

Development and Verification of Local/Global Analysis Techniques for Laminated Composites¹

O. Hayden Griffin, Jr.
Department of Engineering Science & Mechanics
Virginia Polytechnic Institute & State University
Blacksburg, Virginia 24061

Introduction.

Analysis and design methods for laminated composite materials have been the subject of considerable research over the past 20 years, and are currently well developed. In performing the detailed three-dimensional analyses which are often required in proximity to discontinuities, however, analysts often encounter difficulties due to large models. Even with the current availability of powerful computers, models which are too large to run, either from a resource or time standpoint, are often required.

There are several approaches which can permit such analyses, including substructuring, use of superelements or transition elements, and the global/local approach. This effort is based on the so-called "zoom" technique to global/local analysis, where a global analysis is run, with the results of that analysis applied to a smaller region as boundary conditions, in a many iterations as is required to attain an analysis of the desired region.

Before beginning the global/local analyses, it was necessary to evaluate the accuracy of the three-dimensional elements currently implemented in the CSM Testbed. It was also desired to install, using the Experimental Element Capability, a number of displacement formulation elements which have well known behavior when used for analysis of laminated composites.

OBJECTIVES

- **To implement VPI 3-D element family**
- **To define modeling requirements of laminated composites**
- **To develop Global/Local analysis procedure**

OBJECTIVES

This project has three primary objectives:

1. To implement the family of 3-D elements which have been in use for analysis of composite materials at Virginia Tech for the past 15 years (e.g. see References 1-3)
2. To define the special modeling requirements which exist in laminated composites. This research will provide a consistent set of techniques for 3-D analysis of laminated composites.
3. To develop global/local analysis procedures for laminated composites. The development of procedures for both global and local analyses as well as rules for determining proper location of the global/local interface in laminates are included in this research.

This effort will utilize the CSM Testbed exclusively, except where verification of new capabilities must be checked against previously verified software. The Testbed has been in use at Virginia Tech for this project for approximately one year. All numerical results to date have been generated on a DEC VAX 11/780 computer.

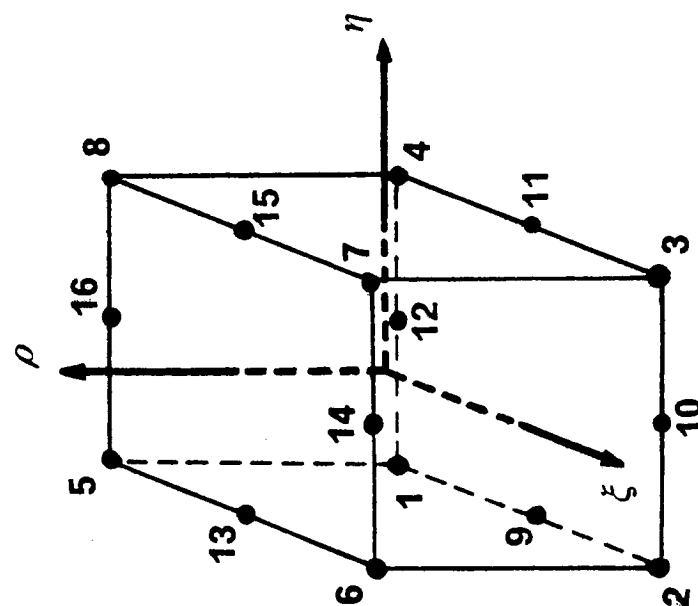
3-D ELEMENTS

- **Implemented using Experimental Element Capability of CSM Testbed**
- **Displacement Formulation**
- **16-, 20-, and 32-node elements implemented**
- **24-node element implementation in progress**
- **Elements well proven by use in NASA-VA TECH Composites Program**

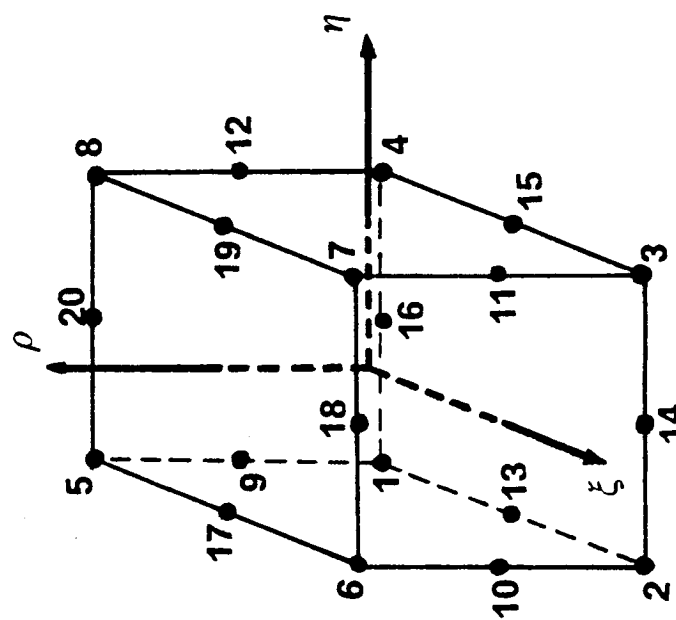
3-D ELEMENTS

Considerable progress has been made to date in enhancing the 3-D analysis capabilities of the CSM Testbed. The approach has been to implement new elements using the Experimental Element Capability of the Testbed. The new elements installed are displacement formulation isoparametric elements sometimes referred to as the "Serendipity" elements. Elements with 16, 20, and 32 nodes have been incorporated, and work is progressing on installation of a 24-node element. All of these elements have orthotropic material properties. They have been used extensively for analysis of composites in the research programs at Virginia Tech for a number of years. Their performance is well understood, and there is a considerable trust in the results obtained due to comparison with other numerical techniques [2] as well as experimental data [4]. In addition to linear elastic constitutive relationships currently being considered for this project, various nonlinear material models, including orthotropic plasticity [2] and an endochronic model [5], have also been developed for these elements.

ELEMENT GEOMETRY



16-Node Element



20-Node Element

ELEMENT GEOMETRY

Two of the element geometries are shown. The 16-node element has eight quadratic edges and four linear edges, while the 20-node element is fully quadratic. Note that the Serendipity elements do not have face nodes or body nodes as do the Lagrangian elements. The 24-node element is similar to the 16-node element in that it has four linear edges. The other eight edges, however, are cubic. The 32-node element is fully cubic. Experience with the 16- and 24-node elements has shown that large inplane-to-out-of-plane aspect ratios can be handled by orienting the element such that the linear edges are in the through-the-thickness direction of the laminate. This is important, since laminates typically are many orders of magnitude larger in planform than in thickness. The element stiffness matrices are evaluated by Gaussian quadrature at a number of Gauss points which is user selectable.

MODELING ISSUES

- **Element selection**
- **Number of elements per ply or fiber orientation**
- **Aspect ratio**
- **Location of 2-D/3-D transition region**

MODELING ISSUES

A number of modeling issues arise in the 3-D analysis of laminates. Some of these issues are unique to composites, while others are always encountered in finite element analysis. They include:

1. Decision of which element to use. The Testbed, in its current form, offers the choice of either hybrid stress or hybrid strain formulation elements in addition to the displacement elements which have been introduced. The choice of elements from the library of a general purpose program has been, is, and will probably continue to be at the discretion of the user.
2. If the laminate is to be modeled such that the interlaminar stresses are determined, each ply, or at least each group of plies with the same orientation, must be subdivided into some number of elements through its thickness. The number of elements through the thickness is of concern. The proper number of elements through the thickness is highly dependent on the order of the element.
3. Since the planar dimensions of a laminate are typically several orders of magnitude greater than the thickness dimension, element aspect ratio will undoubtedly be a problem when conducting 3-D analyses.
4. The location of the global/local interface must be determined. Previous investigators have reported differences in stress prediction due to different interface locations (e.g. [6]). If global/local analysis is to become routine, specific guidelines and techniques for location of an optimum interface position must be determined.

ELEMENT SELECTION

- **Based on problem to be solved**
- **Curved surfaces not easily modeled with linear elements**
- **Doubly curved surfaces cannot be easily modeled with elements with any linear edges**
- **Requirements for elements with highly curved faces may require Lagrangian elements**
- **Special requirements imposed by woven or braided composites**
- **Thin and thick sections may require different elements due to differences in acceptable aspect ratio**

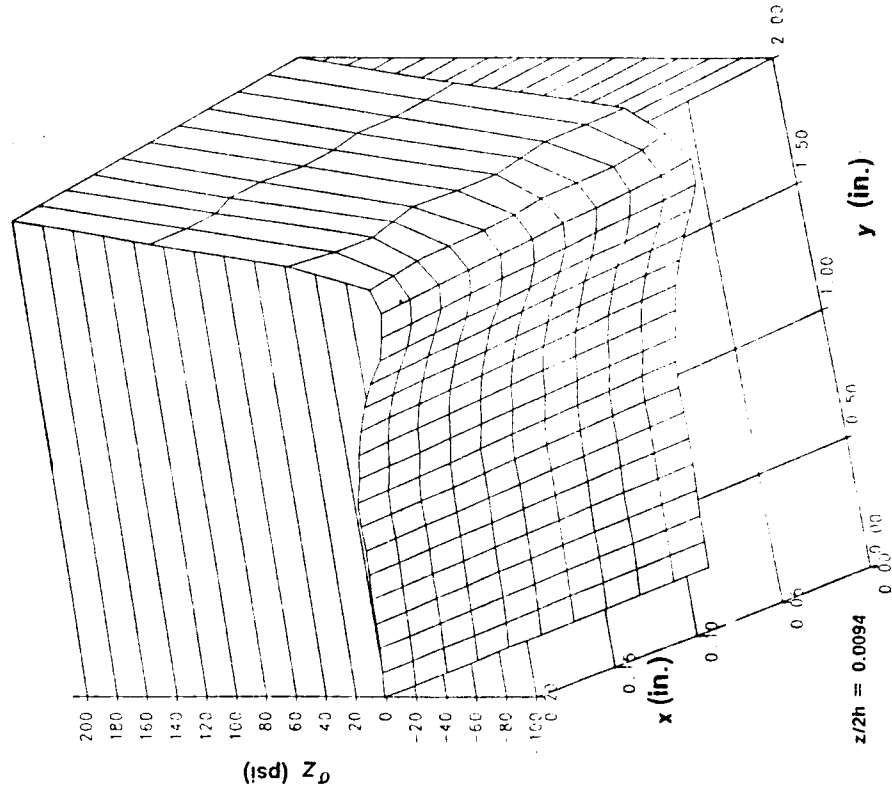
ELEMENT SELECTION

Element selection is based on a number of factors. They include:

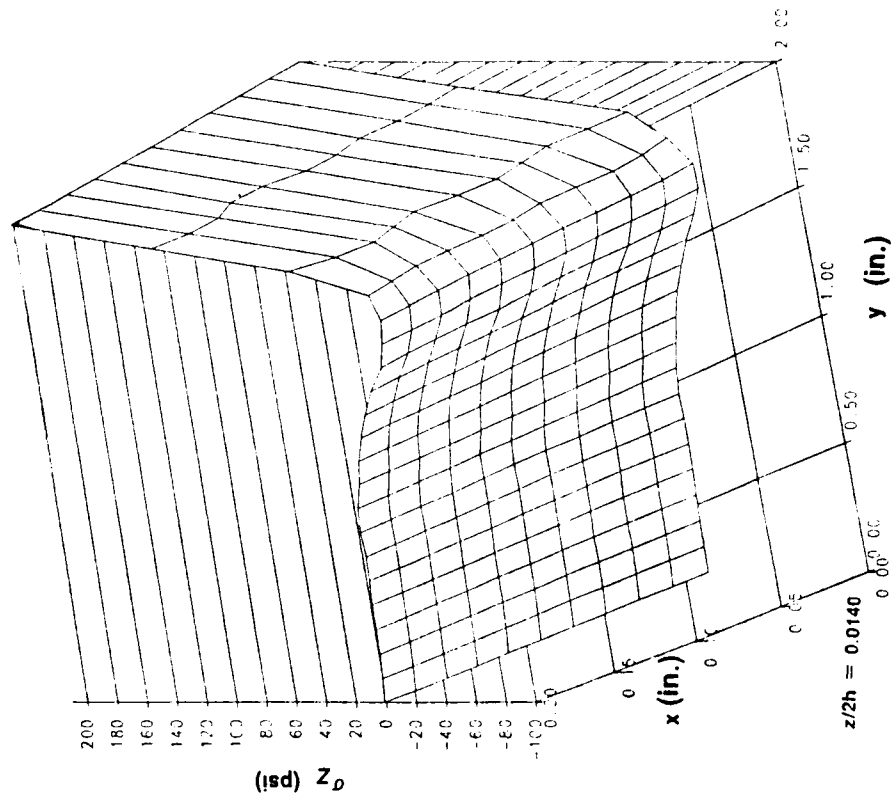
1. The problem which is being solved. The choice of elements for a particular analysis usually depends on the nature of the problem, including geometry, material characteristics, and loading. A grid which might be completely acceptable for analysis of a structure or component under one loading condition might be unacceptable for a different loading. This choice is normally based on the experience of the analyst.
2. Some problem characteristics preclude the use of certain elements for a particular analysis. For example, it is not easy to model accurately a curved surface with elements which have linear edges. Approximation of curves by a series of straight lines may sometimes be acceptable, but this is often not the case. The use of elements which have some linear edges may be possible.
3. If the surface is doubly curved, it may not be possible to use elements which have any linear edges, thus ruling out the use of elements which have a linear geometric shape function on any edge.
4. If the surface of a component is highly curved, the Serendipity elements may not be sufficient for approximation of the body. It may be necessary to use the Lagrangian elements, which have face nodes and can therefore more closely approximate curved surfaces.
5. If composite materials of concern are woven, braided, or filament wound in certain configurations, additional requirements may be imposed on the elements. It may be necessary to use a formulation which allows changes in the fiber angle within an element. This technique is well known [7] and can be implemented in the newly-installed elements, but imposes additional input requirements on the user of specifying exactly how the fiber angle varies.
6. If the geometry of a structure changes drastically, such as a transition from a very thin section to a thick section, it may be necessary to use a mix of element types within the model. Again, this is primarily a decision forced on the analyst, based on experience.

STUDY OF NUMBER OF ELEMENTS PER PLY

σ_z Near Midplane, [0/90], Gr/Ep
Uniform x - direction Extension
20 Node Element, 3 Elements / Angle
ANISAP



σ_z Near Midplane, [0/90], Gr/Ep
Uniform x - direction Extension
20 Node Element, 2 Elements / Angle
ANISAP



STUDY OF NUMBER OF ELEMENTS PER PLY

The reason that a number of elements must be used through the thickness of a particular ply or group of plies at the same angle is the existence of large gradients of stress within the laminate. Some determination must be made of the proper number of elements to use through the thickness, based on a trade-off between accuracy in modeling the gradients and problem size, which becomes substantially larger for every layer of elements which is added. Experience has shown that three linear elements or two quadratic elements through the thickness of each ply angle is sufficient. This is supported by experimental results using Moire interferometry [4]. This slide shows the results obtained for interlaminar normal stress (σ_z) near the midplane for uniform extension in the x-direction of a cross-ply laminate. Note that a smooth distribution is obtained for both two and three elements through the thickness of each ply group. The values are slightly different since the Gauss points for the three elements per layer are slightly closer to the midplane than in the two elements per layer. Examination of the through-the-thickness distribution, however, indicates that either two or three elements per layer gives essentially the same distribution, while the two elements per layer results in a considerably more economical analysis. These results were generated using the finite element program ANISAP [8].

ELEMENT ASPECT RATIO

- **Laminates typically orders of magnitude greater in planar dimensions than in thickness**
- **Aspect ratio difficulties arise immediately when using 3-D elements**
- **Acceptable aspect ratios different for various loading conditions**
- **Element failure is due to ill-conditioning of stiffness matrix**

ELEMENT ASPECT RATIO

It is well known (e.g. [9], [10]) that problems arise when element aspect ratios are not within certain bounds. Ideally, there should be approximately equal strain in all directions for displacement formulation elements [11]. Without a priori knowledge of the solution, however, it is impossible to satisfy that condition. Furthermore, in laminates the planar dimensions of most parts of practical interest are orders of magnitude greater than the thickness. Since a number of elements must be used through the thickness when conducting 3-D analyses, either the element aspect ratio or the number of elements becomes excessive almost immediately. In production analyses, large numbers of elements, with associated large solution time (and cost) are generally unacceptable. Large element aspect ratios result in erroneous results in many cases, most likely due to numerical ill-conditioning of the element stiffness matrix. There is also the potential for an element to become too stiff in one component or the other. The problem is also complicated by the anisotropy of the composite, since for common aerospace materials the ratio of fiber direction modulus (E_1) to transverse direction modulus (E_2) may range from 10 to 100. Hence, optimum inplane aspect ratios may be different than would be expected, due to large differences in strain.

EVALUATION OF EXISTING TESTBED CAPABILITIES

- **S81 (8-node brick) Element**
 - . Uniform extension correct for unidirectional and clustered cross-ply laminates with material and element axes coincident (0 and 90 degree plies)
 - . Problems when material axes not coincident with element axes (45 degree plies) determined to be LAU problem, now fixed.
 - . Acceptable in-plane-to-out-of-plane aspect ratio problem dependent.
- **E43 (plate/shell) element works**

EVALUATION OF EXISTING TESTBED CAPABILITIES

In addition to installing new 3-D elements in the CSM Testbed, work has been done to evaluate the original capabilities, specifically the 3-D brick assumed stress element (S81). The S81 element originally implemented in the Testbed works properly for composite materials so long as the element faces are parallel. The 3-D capabilities of the Laminar Analysis Utility (LAU) were used to generate the property tables for composites. Uniform extension in the x-direction was analyzed for cross-ply and angle-ply symmetric laminates. Original results indicated a problem with the angle-ply laminates. This was later determined to be due to an error in LAU, which has now been corrected. The problem of errors with elements with nonparallel faces has been fixed by NASA Langley CSM personnel. As expected, acceptable S81 element aspect ratios have been determined to be problem dependent. Original results for cross-ply laminates indicated a maximum in-plane thickness ratio of 6:1 before numerical difficulties resulted in a fatal underflow, but other analyses have been run with much larger aspect ratios, with good results.

Evaluation of the E43 element, again using LAU to generate the property tables, has shown this element to be reliable. These analyses were, however, conducted using rectangular elements with reasonable aspect ratios.

GLOBAL/LOCAL ANALYSIS TECHNIQUES

- **Global region**
 - . **Variational (e.g. Rayleigh-Ritz) solutions**
 - . **Analytical solutions**
 - . **Coarse finite element models**
- **Local region**
 - . **Finer grid of 2-D or 3-D elements**
 - . **Substructuring**
 - . **Special elements**
 - . **Displacement and/or slope matching constraints**

GLOBAL/LOCAL ANALYSIS TECHNIQUES

One method of overcoming the previously discussed problems of either large aspect ratios or large problem size is to perform some sort of global/local analysis. There are several techniques for such analyses, both in the global and local regions. These include:

- Global region
 - Variational solutions such as Rayleigh-Ritz have been widely used for solution of plate problems and are a viable choice for the global analysis of plate-type structures. These solutions are well known and trusted.
 - Other analytical solutions may be available for other types of problems. Choice of such models must be made on a case-by-case basis. However, the solution must be valid for the problem of concern, excluding the local effect, which must also be determined to be so local that it does not affect the accuracy of the chosen global model.
 - Coarse finite element models which are accurate for the global response are often chosen for the global region. The use of coarse 2-D models for 2-D problems with local 2-D effects is common. Often a coarse 2-D model, followed by a finer 2-D model, to be followed by a local 3-D model may be used.
- Local region
 - Appropriate choices for a local model may simply be finer discretization of a 2-D or possibly a 3-D finite element model. In that case, the displacements, rotations, etc. from the global model may be applied as boundary conditions on the fine model. One must be careful, however, that the interface between the global and local regions chosen is such that the global model is accurate, that is, the interface must be outside of the locally-affected region.
 - Substructuring, or the use of "superelements," which would be followed by static condensation and then solution of the equations, may be used. This option might be particularly attractive if the software being used offers ready use of this technique.
 - The use of special elements, often with many degrees of freedom, or special "transition" elements is also an alternative. This option, however, requires the software to provide these capabilities, often requiring the developers to have correctly anticipated the classes of problems which will be of interest to users.

RESULTS

- Benchmark of 20-node Experimental Element in Testbed
 - Two rectangular parallelepiped elements with angled boundary face
 - Face angle to 50 degrees gives uniform stress state for uniform force loading
- Uniform extension of cross-ply laminates
 - [0/90]s, [90/0]s, [0/0/90/90]s, and [90/0/90/0]s Gr/Ep Laminates run on CSM Testbed and ANISAP using 20-node displacement element

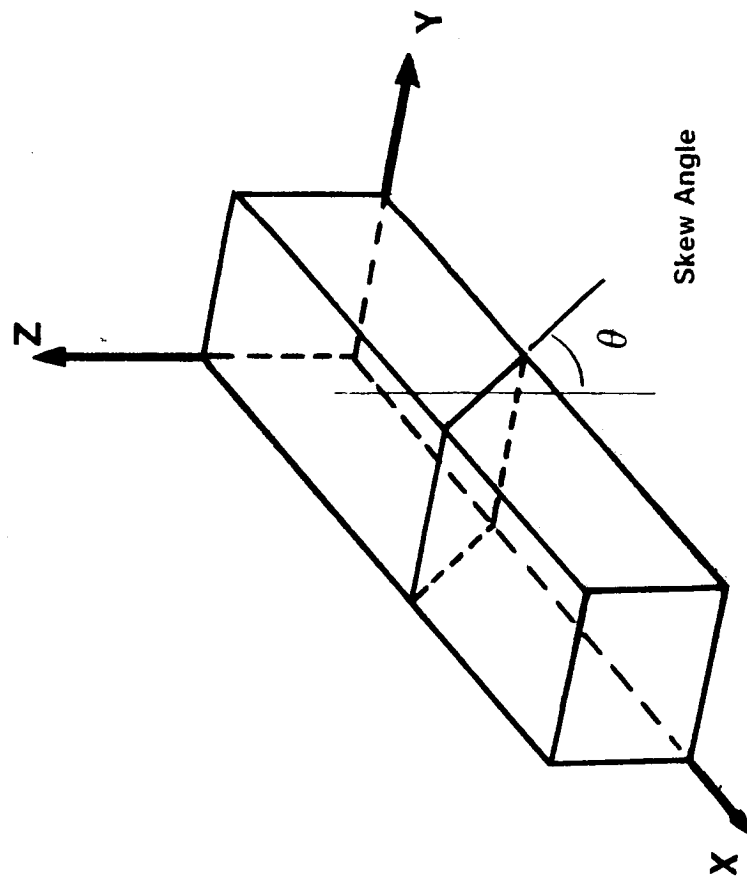
RESULTS

The results to be presented are benchmarking of the experimental displacement elements and for some problems involving uniform extension of cross-ply composites laminates.

For the composite analyses, three different stacking sequences, denoted as $[0/90]_s$, $[90/0]_s$, $[0/0/90/90]_s$, and $[0/90/0/90]_s$ have been considered. Due to geometric, material, and loading symmetry, these are actually two-dimensional problems, with all quantities being independent of the x-coordinate. They have been thoroughly studied and reported in the literature, and are used here as test cases with well known results in order to check the performance of the elements, and serve as a first example of global/local analysis, and to yield results which can be discussed in order to bring the problems associated with analysis of composites into focus before going on to more detailed analyses. Both the CSM Testbed and a VPI program known as ANISAP [8] were used for parts of the analysis. ANISAP uses the 20-node element and has been well checked out, and was thus used as a source of trusted numerical results for comparison with the Experimental Elements which have been implemented in the CSM Testbed.

It should be noted that all of the laminates studied had a half length in the x-direction of 0.2 inch, a half width in the y-direction of 2.0 inches and a half thickness of 0.4 inch. A uniform x-direction displacement of 0.001 inch was applied at the face $x=0.2$, resulting in a uniform x-direction strain $\epsilon_x = 0.005$. For the analyses, the composite material was assumed to be homogeneous and transversely isotropic, with the properties approximately that of T300/5208 graphite/epoxy ($E_1 = 19.2$ msi; $E_2 = E_3 = 1.56$ msi; $G_{12} = G_{13} = 0.83$ msi; $G_{23} = 0.5E_2/(1 + \nu_{23})$; $\nu_{12} = \nu_{13} = 0.23$; $\nu_{23} = 0.4$).

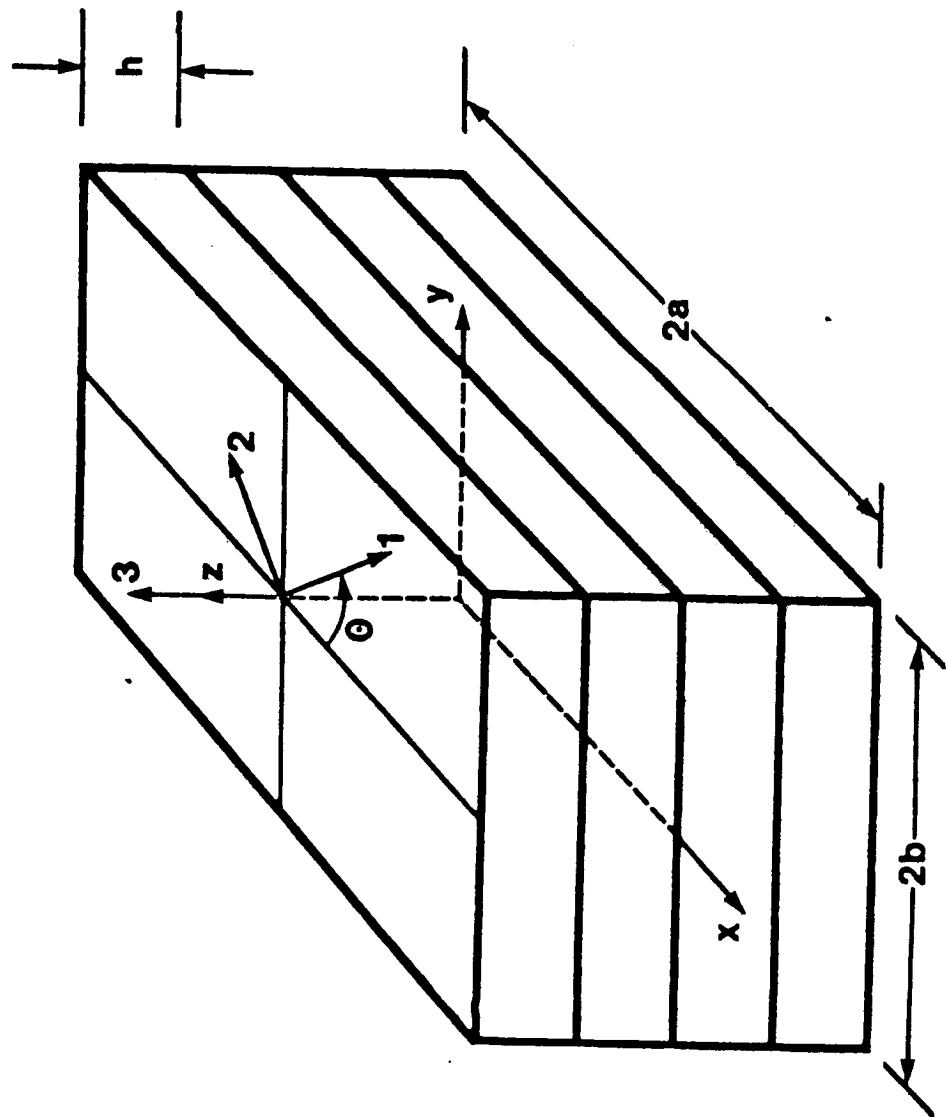
SCHEMATIC OF BENCHMARK



SCHEMATIC OF BENCHMARK

The simple, two element benchmark performed on the 20-node displacement element is shown. The y-z plane is a plane of symmetry, and the node at the origin was fixed to eliminate rigid body rotation. Both uniform x-direction force and displacement have been applied, both of which should result in a uniform x-direction stress, with all other stresses zero. The angle θ was varied from 0° to 50° , and at no time did the stress distribution show any variation. The next benchmark to be run consists of a skewed solid within a cube (MacNeal-Harder solid benchmark, [12]), but this is incomplete at this time.

SCHEMATIC OF LAMINATE

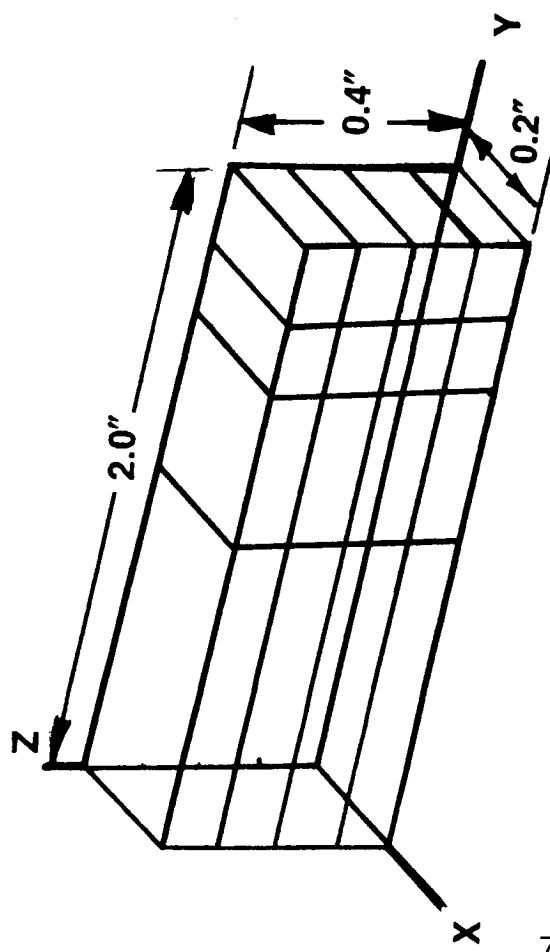


SCHEMATIC OF LAMINATE

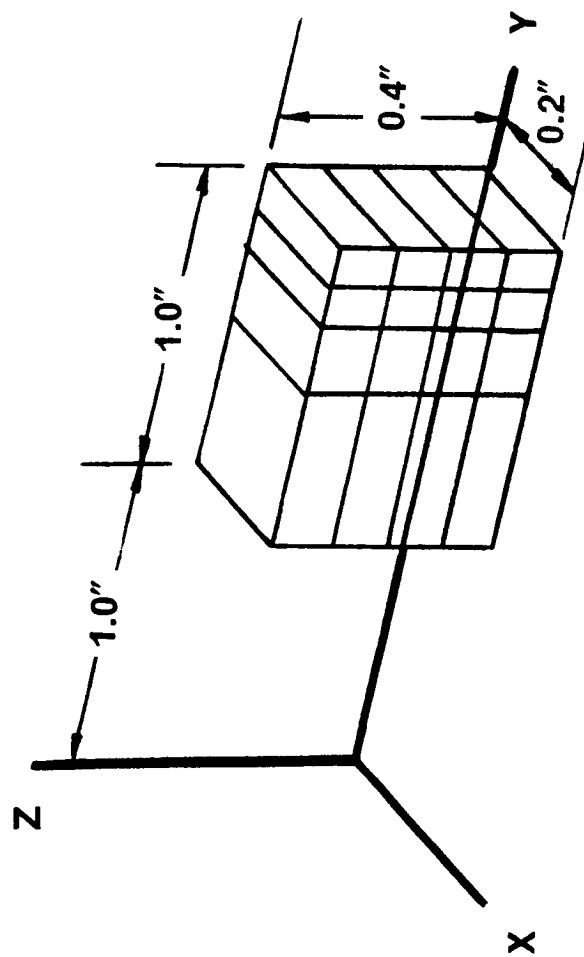
The slide shows a general laminate geometry. The xyz axis system, normally referred to as the "laminar coordinate system," is the system in which loads are applied. The 123 coordinate system, normally referred to as the "material coordinate system," is such that the 1-axis is parallel to the fibers. Thus, the angle θ is a measure of how the fibers are oriented. The 123 system is rotated about the 3 (z) axis for each ply which has a fiber angle (θ) which may be nonzero. The laminate has dimensions $2a$, $2b$, and $4h$ in the x-, y-, and z-directions, respectively.

Note that for the laminates to be discussed here that θ will be either 0° or 90° . For these laminates, each of the coordinate planes is a plane of symmetry, and the problem is actually two-dimensional. It is presented here as a three-dimensional analysis for purposes of illustration. The actual model has three planes of symmetry, achieved by restraining the x-direction displacements on the plane $x=0$, the y-direction displacements on $y=0$, and the z-direction displacements on $z=0$. A uniform x-direction displacement of the face of maximum x-coordinate is achieved by specifying the displacement of all nodes on that face to have the desired value. All other displacements are free.

FE MODELS USED FOR GLOBAL/LOCAL ANALYSIS



Global FE Model



Local FE Model

FE MODELS USED FOR GLOBAL/LOCAL ANALYSIS

The finite element grids shown were used for global and local analyses of the laminates. Each grid consists of two elements per ply orientation, four elements in the y-direction, and one element in the x-direction. Only the 20-node displacement elements were used in the laminate analyses.

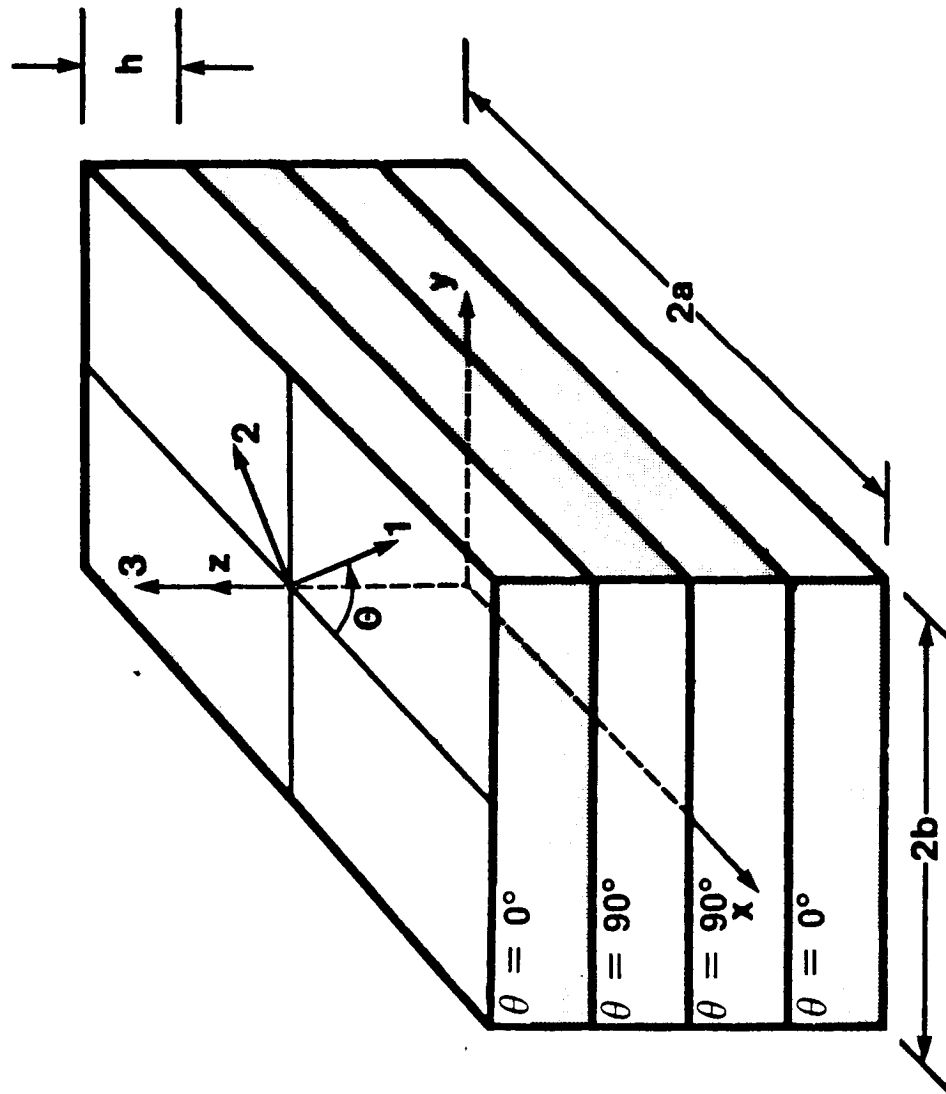
The global model uses the four elements in the y-direction over the complete half width of the laminate. The largest element, in the region where classical lamination theory results will be recovered and the gradients are the smallest, is one-half of the total dimension. The next largest elements are one-half of the remaining y-direction dimension. The remaining elements are of equal size, thus evenly dividing the remaining one-fourth of the dimension.

The local model also has four elements in the y-direction, but covering only one-half of the y-dimension of the modeled part of the laminate. The relative size of the elements is the same as the global model.

For the global/local analyses, the global analysis was first run. The displacements at the plane $y = 1.0$ were then applied to the $y = 1.0$ face of the local model, and the local model was then run. This corresponds to a very simplified application of the zoom technique of global/local finite element analysis.

If it is desired to use a larger number of elements through the thickness in the local model than in the global model, additional constraints are required for the "new" nodal points on the global/local interface. The details of how these constraints are derived are one of the considerations of this project. The simplest method would be to linearly interpolate between "old" nodes, but since the displacements for the 20-node element are quadratic, that may not be appropriate. Several algorithms for determining these displacements are discussed in Reference 13.

SCHEMATIC OF [0/90]s LAMINATE

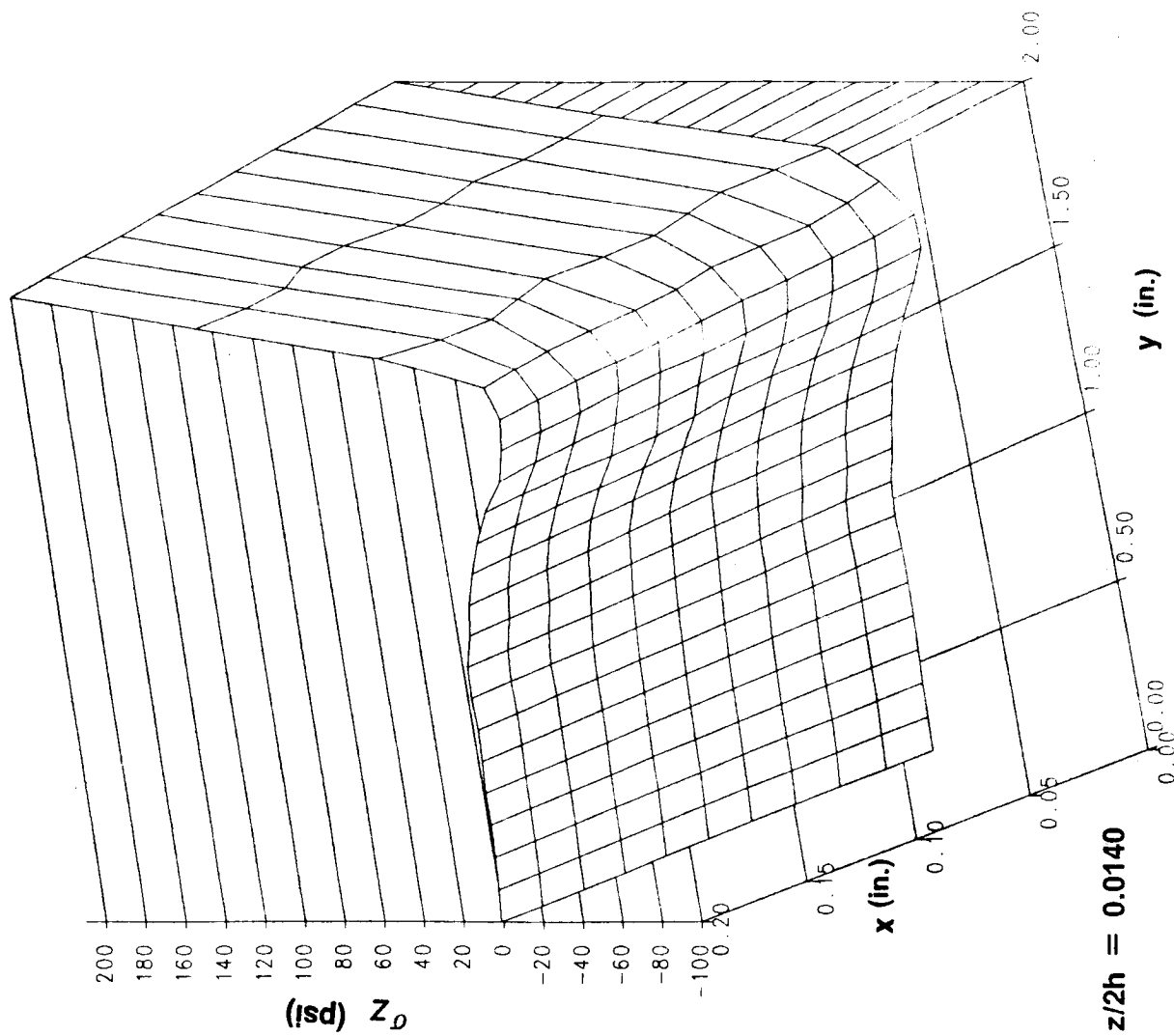


SCHEMATIC OF $[0/90]_s$ LAMINATE

The schematic of the $[0/90]_s$ laminate is shown. The fibers in the upper and lower layers have fibers parallel to the x-axis ($\theta = 0^\circ$). The two inner layers have fibers parallel to the y-axis ($\theta = 90^\circ$). The global finite element model previously shown was used to model the uniform x-direction extension of the laminate.

As previously discussed, the laminate was modeled using two 20-node elements through the thickness of each ply orientation. The x-, y-, and z-coordinate planes are constrained as described.

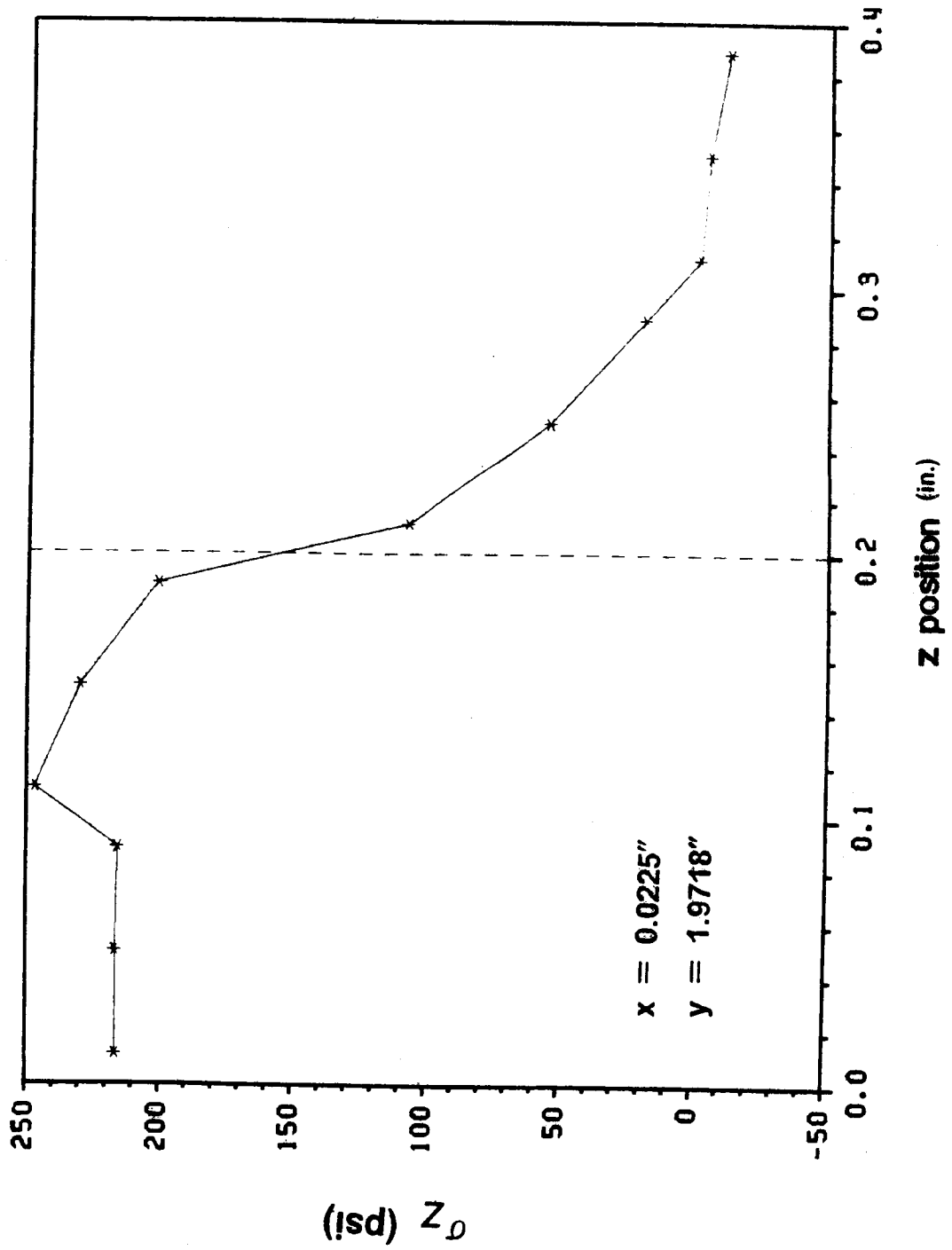
σ_z Near Midplane, [0/90], Gr/Ep
 Uniform x - direction Extension
 20 Node Element, 2 Elements / Angle
 CSM Testbed



σ_z NEAR MIDPLANE, $[0/90]_s$ LAMINATE

The distribution of σ_z near the midplane of the $[0/90]_s$ laminate is shown in this slide. Note that the stress is independent of the x-coordinate, showing a high tensile value near the edge. This stress, known as the interlaminar normal, or "peel" stress is, in fact, often the cause of failure in this laminate loaded in this manner. The region of compression away from the edge is necessary if equilibrium of the top half of the laminate is considered. Since no z-direction forces are applied, if there is a region of tensile σ_z , then there must be a corresponding region of compressive σ_z such that the integral of all z-direction stresses over any plane in the laminate is zero.

Z vs. z, [0/90]_s Graphite/Epoxy
Uniform x - direction Extension
20 Node Element, 2 Elements / Angle
ANISAP

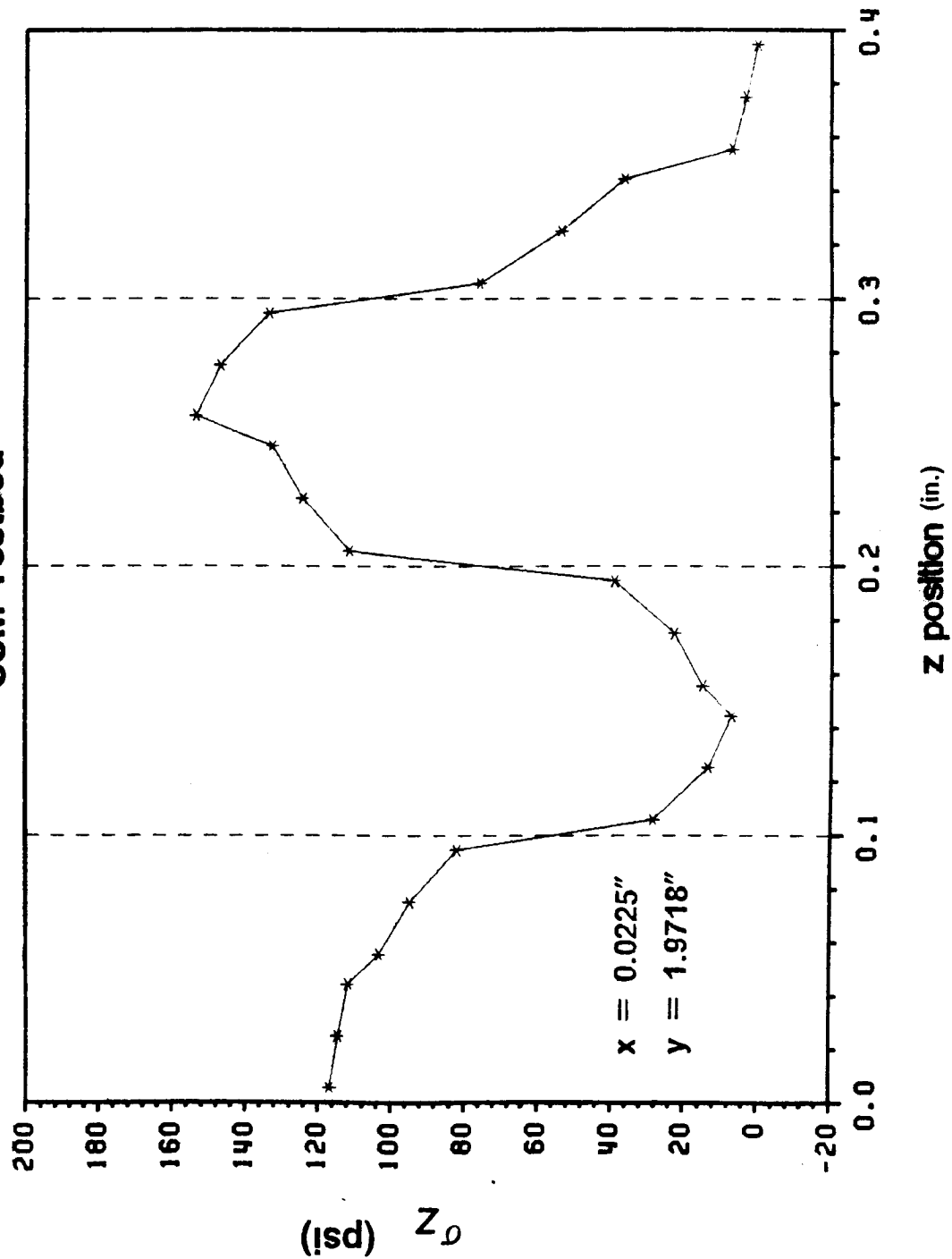


σ_z vs. z , $[0/90]_s$ LAMINATE

The distribution of σ_z through the thickness of the $[0/90]_s$ laminate at the Gauss point of maximum y -coordinate is shown in this slide. Note that the value is maximum slightly away from the midplane, and tends to zero near the top surface, as required by equilibrium. The fact that the stress is not exactly zero at the free surfaces may indicate that more elements are required through the thickness. This is also a problem with displacement elements, which are known to not satisfy equilibrium everywhere. It is possible that the hybrid stress or strain formulation elements of the Testbed may prove superior to the displacement elements for analysis of composites, especially where exact satisfaction of equilibrium is of concern.

Note the finite element prediction of high stress gradients within this laminate. Classical Lamination Theory, in fact, predicts jumps in certain stress components across ply interfaces. In reality, however, jumps in stress are relieved by some dissipative or failure event at the location of the discontinuity. The nature of this phenomenon is governed to a large degree by the characteristics of the matrix, since such effects normally occur at or near interfaces which tend to be resin rich.

**σ_z vs. z , $[0/90/0/90]_s$ Graphite/Epoxy
 Uniform x - direction Extension
 20 Node Element, 2 Elements / Angle
 CSM Testbed**



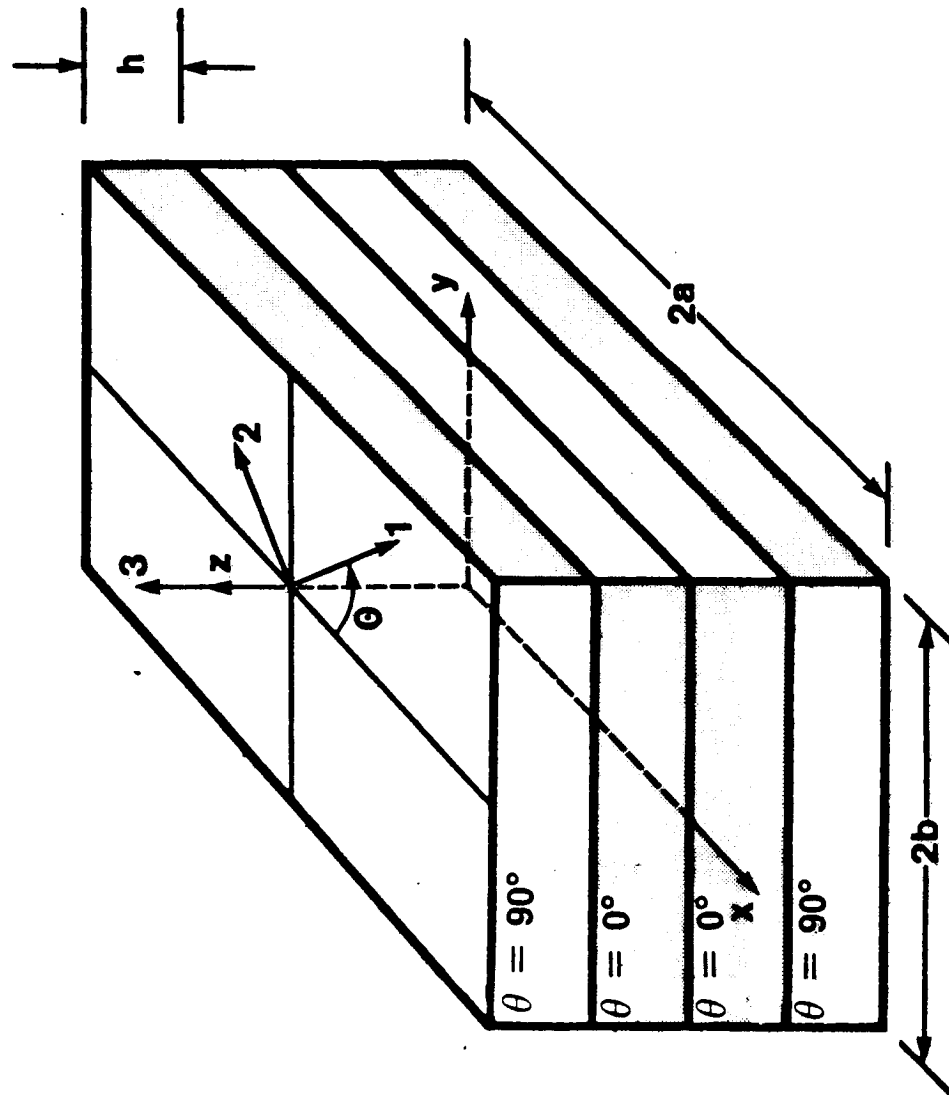
σ_z vs. z, [0/90/0/90]_s LAMINATE

It has been shown experimentally [14] that the ultimate load carrying capability of a cross-ply laminate is increased if the individual plies are arranged such that few layers of equal angle are placed together. In order to examine this situation numerically, an analysis was conducted of a cross-ply laminate of the same thickness as the previous one, but where the plies were arranged such that the thickness of 0° and 90° ply groups was cut in half. This laminate, which could therefore be designated [0/90/0/90]_s, will be referred to as a "staggered" laminate, as opposed to the "clustered" [0/90]_s laminate. For laminates of equal thickness, same material and loading, but with the different stacking sequence, there should be some indication that σ_z is lower in the "staggered" laminate.

The distribution of σ_z through the thickness of the [0/90/0/90]_s laminate at the Gauss point of maximum y-coordinate is shown in this slide. Note that the value is not maximum near the midplane as with the clustered laminate, but is maximum in the interior. Also note that the maximum value, as expected, is substantially lower than the clustered laminate. As before, the stress tends toward zero at the top surface of the laminate, as required by equilibrium.

There are also high stress gradients within the staggered laminate. The high gradients are to be expected in laminates, and an analyst should expect them and design the finite element model accordingly. One should also be aware that laminate stacking sequence plays an important role on the magnitude and distribution of stress within the laminate.

SCHEMATIC OF [90/0]s LAMINATE

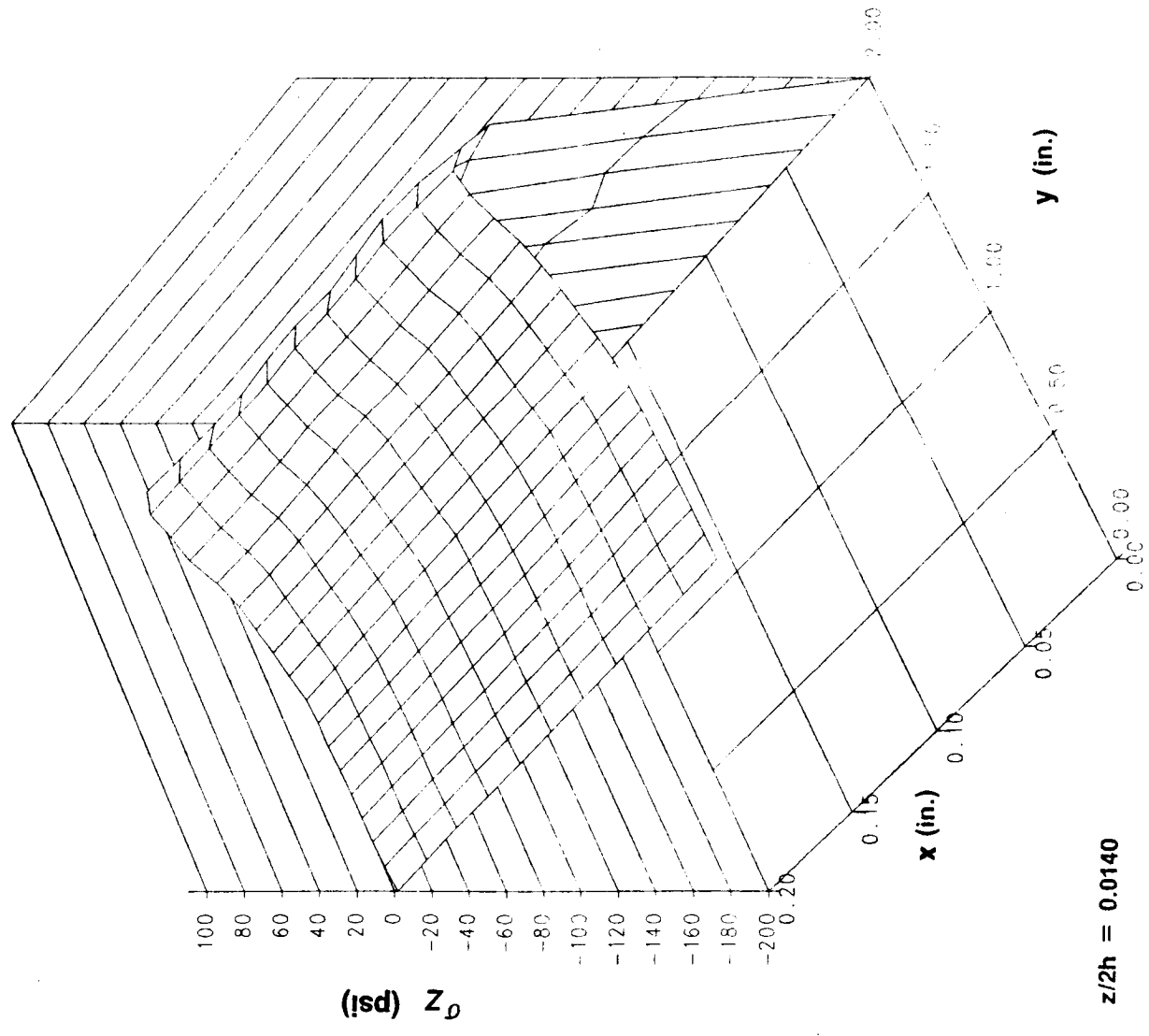


SCHEMATIC OF $[90/0]_s$ LAMINATE

The schematic of the $[90/0]_s$ laminate is shown. The fibers in the upper and lower layers have fibers parallel to the y-axis ($\theta = 90^\circ$). The two inner layers have fibers parallel to the x-axis ($\theta = 0^\circ$). The global finite element model previously shown was used to model the uniform x-direction extension of the laminate.

ORIGINAL PAGE IS
OF POOR QUALITY

σ_z Near Midplane, [90/0], Gr/Ep
Uniform x - direction Extension
20 Node Element, 2 Elements / Angle
ANISAP

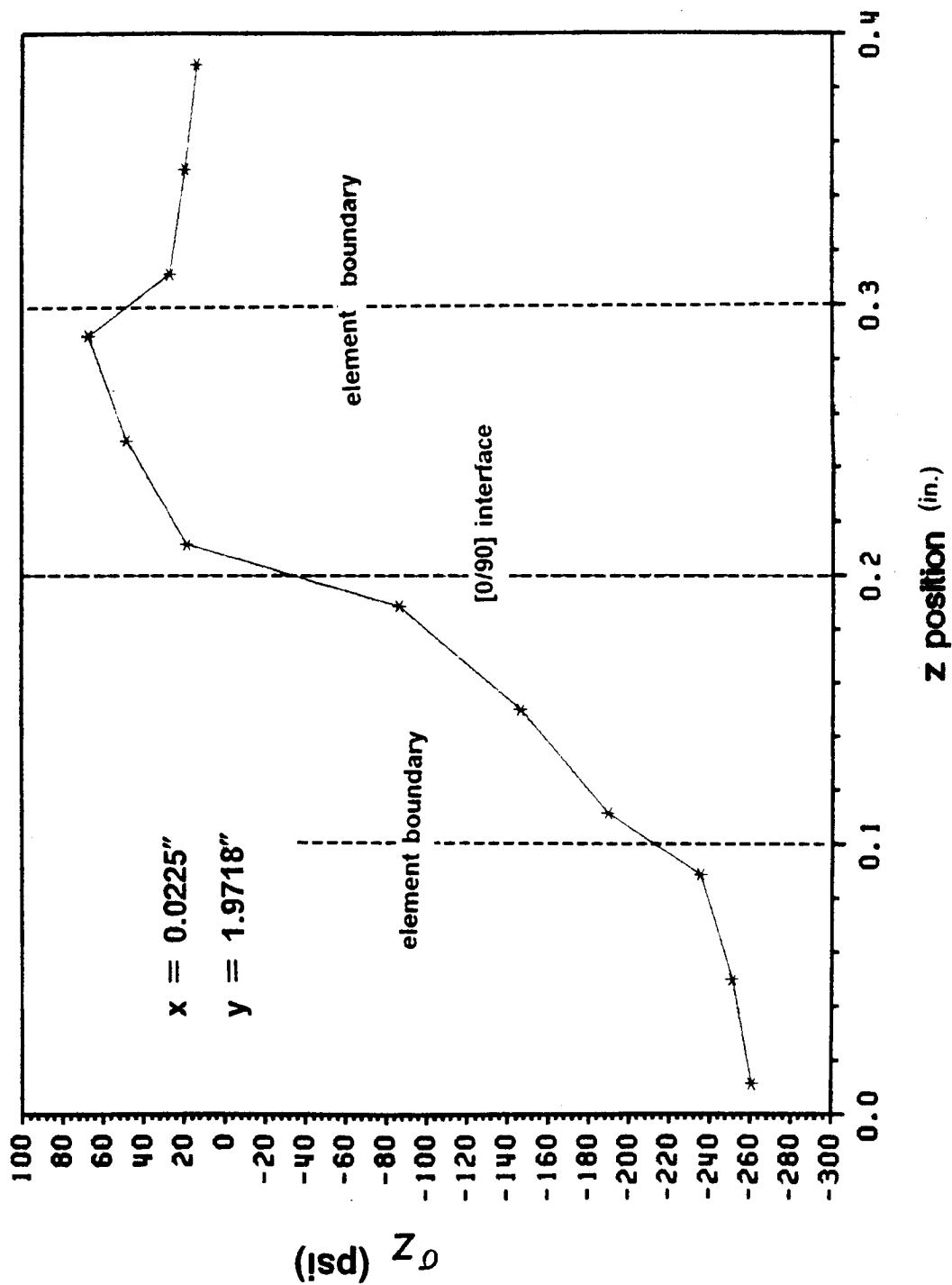


$z/2h = 0.0140$

σ_z NEAR MIDPLANE, $[90/0]_s$ LAMINATE

The distribution of interlaminar normal (σ_z) near the midplane of the $[90/0]_s$ laminate is shown. Although it appears to be a mirror image of the distribution of the $[0/90]_s$ laminate, it is not, as can be seen by comparison of the stress values. As before, however, note that any region of nonzero σ_z requires a region of σ_z with opposite sign. For this laminate the large value of σ_z is compressive, with an absolute value approximately five times the maximum tensile value. Since a tensile σ_z is the component which results in delamination type failures, it may be concluded that this layup would have little tendency to delaminate under the given load.

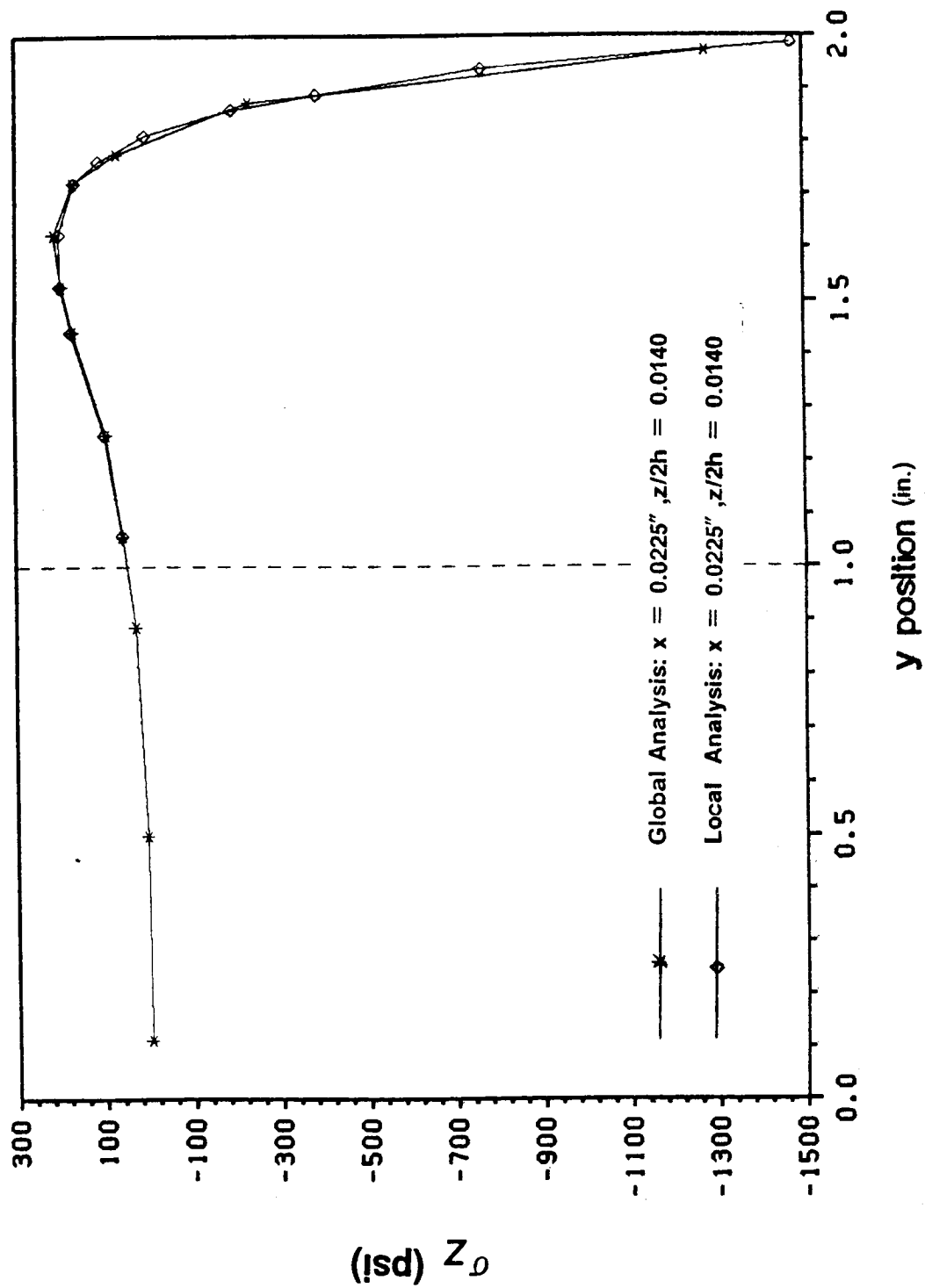
σ_z vs. z , $[90/0]_s$ Graphite/Epoxy
 Uniform x - direction Extension
 20 Node Element, 2 Elements / Angle
 ANISAP



σ_z vs. z , $[90/0]_s$ LAMINATE

The distribution of σ_z through the thickness of the $[90/0]_s$ laminate at the Gauss point of maximum y -coordinate is shown in this slide. Note that the compressive value is maximum near the midplane and tends toward zero at the top surface of the laminate, as required by equilibrium.

σ_z Near Midplane, $[90/0]_s$ Gr/Ep
 Uniform x - direction Extension
 20 Node Element, 2 Elements / Angle
 CSM Testbed, Global - Local Analysis

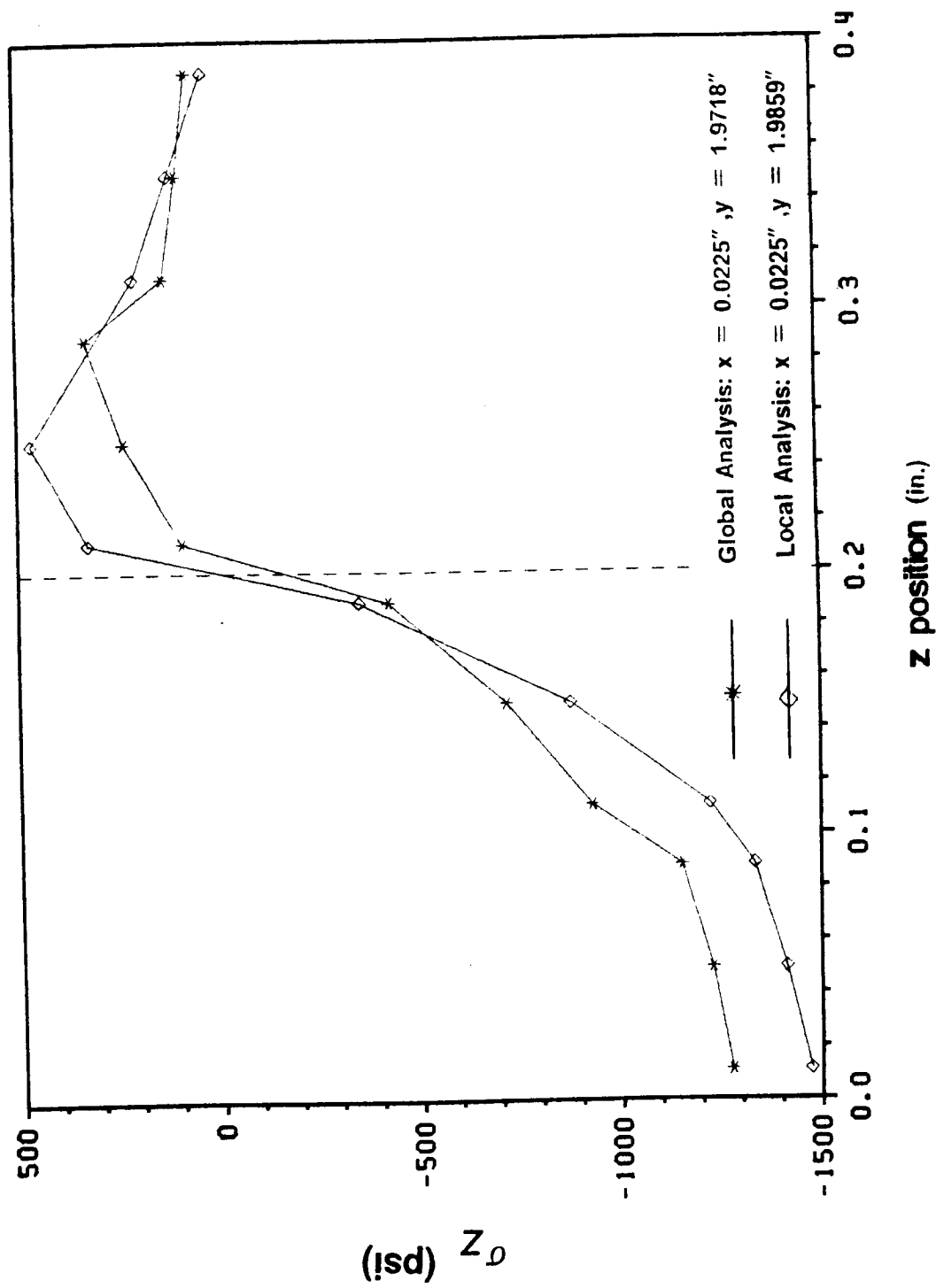


σ_z vs. y NEAR MIDPLANE, [90/0]_s, GLOBAL/LOCAL ANALYSIS

The [90/0]_s laminate was subjected to the global/local procedure as previously described, by analyzing the laminate using the global model, then applying the displacements calculated on the plane $y = 1.0$ to the $y = 1.0$ face of the local model. The predicted values of σ_z vs. the y -coordinate, near the midplane, are shown for the global model (* symbols) and the global/local model (square symbols). Note that there is excellent agreement between the models, thus indicating that the global model was accurate over its range. Also note, however, that the Gauss point nearest the edge of the global/local model is nearer the edge than for the global model, and that the stress at that point is higher than the global model. In analyses of this type, the region of maximum stress is of most interest, and the analyst is always trying to push the solution toward that point. The global/local model appears to provide an accurate means of doing so, while being more economical than a finer global model.

While this is obviously a simplified example of a global/local analysis, it illustrates the application and utility of the technique.

σ_z vs. z , $[90/0]_s$ Graphite/Epoxy
 Uniform x - direction Extension
 20 Node Element, 2 Elements / Angle
 CSM Testbed, Global - Local Analysis



σ_z vs. z , $[90/0]_s$ LAMINATE, GLOBAL/LOCAL ANALYSIS

The distribution of σ_z through the thickness of the $[90/0]_s$ laminate at the Gauss point of maximum y -coordinate for the global and global/local models is shown in this slide. Note that the global/local model yields a higher compressive stress near the edge, as expected. It also appears that the tensile stress is higher than predicted by the global model, and at a slightly different location in the laminate. This may not be a disagreement between the models, however, since the Gauss points at which the global/local stress is plotted is closer to the edge than for the global model.

CONCLUSIONS

- Installing Experimental Elements in CSM Testbed is straightforward once you know the rules
- Once installed in the Testbed, new elements are readily benchmarked
- As previously shown experimentally, laminates with "staggered" plies (e.g., $[0/90/0/90]_s$) exhibit substantially lower interlaminar stresses than laminates with "clustered" plies (e.g., $[0/0/90/90]_s$)
- First runs of 3-D Global/Local analysis using 20-node displacement elements in Testbed were successful. Results compare with proven software

CONCLUSIONS

A number of conclusions, both on the use of the CSM Testbed and the mechanics of laminated composites, can be drawn from the work conducted to date. They include:

1. Installing new elements in the CSM Testbed using the Experimental Element Capability is straightforward once you know all of the rules. The documentation on this capability is somewhat fragmented and may be incomplete. It appears that the Generic Element Processor, which will be in the new release of the Testbed, will be of great utility in efforts such as the present study. It provides an even easier means of installing new elements for experimentation and use, as well as providing geometric nonlinearity via the corotational technique. It appears that in an educational environment, where the time a student is actually working on a degree may be the limiting factor, this capability will allow researchers to readily implement new elements and then spend more time testing and using the elements rather than divert valuable time toward the development of complete new software implementations before testing and use can begin. This is very valuable to those of us who are more interested in developing new capabilities and then immediately using them to study the problems which are of real interest to us, as opposed to those who are more interested in developing the new capabilities and then turning them over to others for use.
2. Once installed in the Testbed, new elements are readily checked out, benchmarked, and ready to use. There is no additional time required to test the validity of assembly routines, solvers, etc., since those are provided by the Testbed.
3. It has been shown that cross-ply symmetric laminates with "staggered" plies have lower interlaminar normal stresses due to uniform extension than do laminates with "clustered" plies. This confirms experimentally observed behavior.
4. Simple global/local finite element analysis using coarse and fine grids of 20-node displacement elements yielded good results. While the global model appears to be accurate, use of the global/local model enables prediction of stresses nearer the edge than with the global model.

PLANS

- . Use "zoom" technique for local/global analyses**
- . Solve problem of pure bending of composite plate with central circular hole using CLT and/or E43 elements for global solution and 3-D FEA for local solution**
- . Add stiffener(s) with at least one interrupted by hole**
- . Consider stress distributions around stiffener/hole/plate junctures**
- . Determine guidelines for location of local/global interface, semi-automatic if possible**

PLANS

The zoom [13] technique will be used for global/local analysis. This technique offers a straightforward method which allows all analyses to be done with the Testbed. It is likely that either the E43 element of the Testbed or one of the new formulations installed via the Generic Element Processor will be used for any 2-D analyses which are required. Investigation will also be made into the use of classical lamination theory results for generating strains and curvatures, which can then be converted to displacements and applied to FE models.

The target problem is the CSM Focus Problem, a composite plate with blade stiffeners, with at least one stiffener interrupted by a hole. Of primary interests in the composite mechanics are the discontinuities, the regions near the hole and the juncture of the stiffeners and the plate. Detailed 3-D analyses will be conducted in these regions in order to understand the mechanism of load transfer and hence failure.

In addition to rules for 3-D analysis of laminates, guidelines will be developed which can be used for the corresponding global/local analyses. These will include guidelines for estimating the proper location of the global/local interface, and for assessing the validity of solutions once that decision has been made. An attempt will be made to develop semi-automatic techniques for location of the interface and also for examination of the solution to issue a report to the user which would point out any regions where the solution might be questionable.

REFERENCES

1. R. M. Barker, F. T. Lin, and J. R. Dana, "Three-dimensional Finite-element Analysis of Laminated Composites," *Computers and Structures*, Vol 2, 1972, pp 1013-1029.
2. O. H. Griffin, Jr., M. P. Kamat, and C. T. Herakovich, "Three-dimensional Inelastic Finite Element Analysis of Laminated Composites," *Journal of Composite Materials*, Vol 15, Nov 1981, pp 543-560.
3. S. W. Burns, C. T. Herakovich, and J. G. Williams, "Compressive Failure of Notched Angle-Ply Composite Laminates: Three Dimensional Finite Element Analysis and Experiment," VPI&SU Report VPI-E-85-22 (CCMS-85-11), Nov 1985.
4. C. T. Herakovich, D. Post, M. B. Buczek, and R. Czarnek, "Free Edge Strain Concentrations in Real Composite Laminates: Experimental-Theoretical Correlation," VPI&SU Report VPI-E-84-24 (CCMS-84-24), Jul 1984.
5. S. R. Mathison, C. T. Herakovich, M. J. Pindera, and M. J. Shuart, "Nonlinear Analysis of the Response and Failure of Compression-Loaded Angle-Ply Laminates with a Hole," VPI&SU Report VPI-E-87-14 (CCMS 87-12), Jul 1987.
6. C. T. Sun and P. Mehrotra, "Global-Local Structural Analysis Using a Multiprocessor Computer," in the proceedings of the NASA Langley CSM Review, August 25, 1986, W. J. Stroud, Editor.
7. E. L. Stanton, L. M. Crain, and T. F. Neu, "A Parametric Cubic Modelling System for General Solids of Composite Material," *International Journal for Numerical Methods in Engineering*, Vol 11, 1977, pp 653-670.
8. S. W. Burns, S. R. Mathison, and C. T. Herakovich, "ANISAP--A Three Dimensional Finite Element Program for Laminated Composites Subjected to Mechanical Loading," VPI&SU Report VPI-E-86-12 (CCMS 86-03), Jul 1986.
9. R. J. Melosh, "Development of the Stiffness Method to Define Bounds on Elastic Behavior of Structures," Ph.D. Dissertation, University of Washington, 1962.
10. F. T. Lin, "The Finite Element Analysis of Laminated Composites," Ph.D. Dissertation, Virginia Polytechnic Institute & State University, 1972.
11. C. S. Desai and J. F. Abel, Introduction to the Finite Element Method, Van Nostrand Reinhold, New York, 1972.
12. R. H. MacNeal and R. L. Harder, "A Proposed Standard Set of Problems to Test Finite Element Accuracy," *Finite Elements in Analysis and Design*, Vol 1, 1985, pp 3-20.
13. A. K. Noor, "Global-Local Methodologies and Their Application to Nonlinear Analysis," *Finite Elements in Analysis and Design*, Vol 2, 1986, pp 333-346.
14. C. T. Herakovich, "Influence of Layer Thickness on the Strength of Angle-Ply Laminates," *Journal of Composite Materials*, Vol 16, May 1982, pp 216-227.